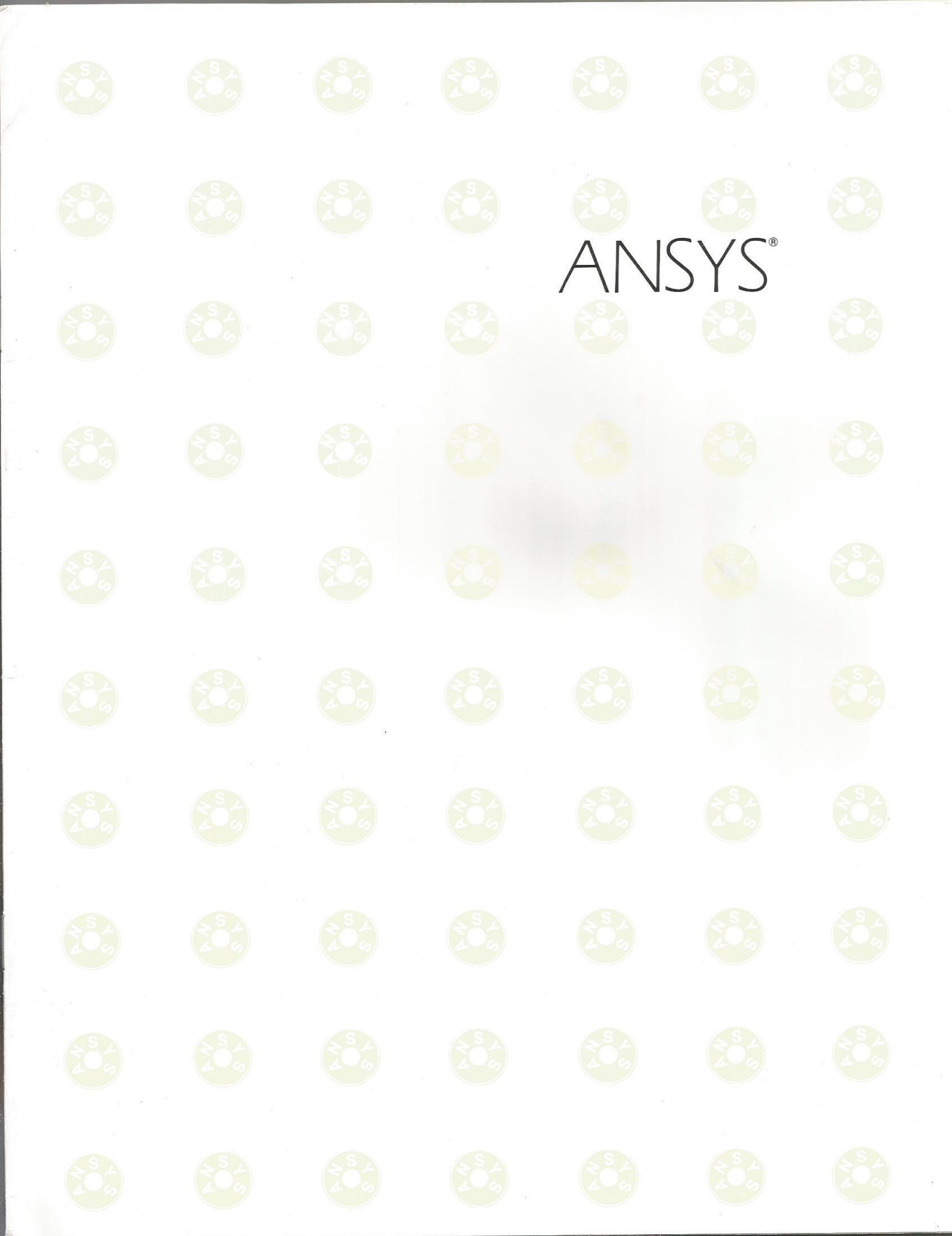


A grid of small, light green circular watermarks is overlaid on the page. Each watermark contains the text 'ANSYS' in a stylized font, with the 'A' and 'S' being larger and more prominent than the 'N' and 'Y'. The grid is composed of approximately 10 columns and 10 rows of these circular logos.

ANSYS®

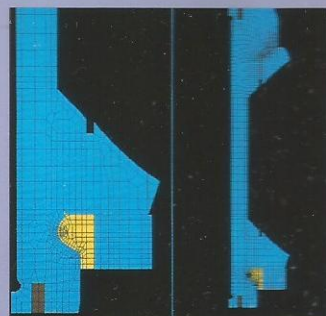


...the fragile lily pad still manages to resist being swept away. How? Attached to the bouyant pad, a long, flexible stem anchors the lily securely in place, letting it absorb motion. It's a principle as old as nature itself: to bend instead of breaking.

Today, thanks in large part to ANSYS, the widely-recognized finite element computer program for engineering analysis, this same principle is being used in one of history's most ambitious engineering projects—the deepwater tension leg drilling platform (TLP).

Like a 53,500-ton lily pad, the TLP floats on the surface of the North Sea, secured against 100-foot waves and 100+ MPH winds by a series of tethering lines and flexible couplings moored to the sea floor nearly 500 feet below. Key to its success are the flexible couplings which protect the tethering lines from fatigue while controlling lateral motion and preventing vertical motion of the TLP. In designing these couplings, Leslie Associates used ANSYS to evaluate a wide range of design possibilities. They analyzed stresses and displacements throughout the metal components, performed max-min tolerancing checks, and analyzed friction in gap elements between mating surfaces to arrive at the best possible design.

Of course, having the ability to explore the full spectrum of design possibilities can be very important for you, too. Because whether you're analyzing drilling platform components or designing structures, machine parts, or consumer products, ANSYS can help you make almost any product better and more profitably. By helping you find the best way to make it.



ension leg drilling platform
flexible coupling

A COMPANY AND A COMPREHENSIVE FEA PROGRAM THAT HAVE CHANGED WITH THE TIMES.

Like many software developers, Swanson Analysis Systems, Inc. (SASI) started out as a small group of talented people working out of a private home. But we've come a long way since then. Through the years, as ANSYS has gained worldwide recognition and acceptance, SASI has grown and expanded. Today we employ more than 70 people at our headquarters. What's more, we operate an international network of ANSYS Support Distributors who can provide service to customers virtually anywhere on earth.

There are good reasons why ANSYS is one of the most widely-recognized large-scale, general purpose finite element programs for engineering analysis available today. Since 1970, when Swanson Analysis Systems, Inc. began operations, the ANSYS program has undergone continuous revision and upgrading to keep pace with the rapid evolution of computer hardware. This ongoing series of enhancements ensures that ANSYS' uses and capabilities stay current with and often anticipate the needs of the marketplace. At SASI, we're committed to technological leadership in finite element analysis (FEA).

When it was first developed for use by the power generation and metals industries, ANSYS was a general purpose, batch FE program for mainframe computers that included heat transfer and structural analysis. During the next several years, capabilities including nonlinear effects, substructuring, and isoparametric elements were added. In the mid-1970's, SASI began exploring uses of vector graphic terminals and the new minicomputers. And two important capabilities, the interactive mode of operation and a preprocessor for automatic mesh generation, were added in the late 1970's.

By 1981, ANSYS offered more sophisticated elements such as the 3-D plastic beam, new solution options such as reduced subspace iteration, and on-line documentation. Interactive graphics were incorporated as an important part of the program to verify geometry, materials, and boundary conditions. In time, these interactive graphics were upgraded to vector or full 3-D color raster fill. SASI consolidated the interactive

postprocessing, thereby offering an alternative to the batch mode of structured output. The most recent ANSYS revisions include design optimization features, preprocessing data base storage, solid modeling, and an extended graphics library with provisions for animation, light source shading, hidden surface removal, 3-D transformations, and other graphic display options. We also continue to explore the capabilities of new hardware such as parallel processors, microcomputers, and enhanced graphics display devices. Future ANSYS updates will continue to use the most advanced proven technology available to provide our users with the finest in FEA capabilities.

WE PIONEERED FEA QUALITY ASSURANCE. AND WE'RE STILL THE LEADERS.

An FEA program is only as useful and valuable as it is reliable, so as ANSYS has changed and developed through the years, our quality assurance program has evolved right along with it. At SASI, we believe that maintaining the high quality of the ANSYS program is of paramount importance. Because when our program performs the way you want it to, it makes your job that much easier.

ANSYS is based on proven engineering concepts that make it a solid FEA program—and one that's continually verified by SASI. Our Quality Assurance (Q.A.) program began as a limited series of internal verification problems designed to test the efficiency and accuracy of the ANSYS program's operation. Today, SASI employs a full-time quality assurance staff and the ANSYS program goes through an exacting Q.A. procedure including more than 2000 verification

problems before being approved for release. SASI engineers compare ANSYS-calculated solutions with known theoretical solutions and results calculated in the previous ANSYS release. All to ensure that you have a dependable and efficient FEA program.

Our diligence and attention to detail have paid off. The first ANSYS Verification Manual, published in 1976, was one of the first manuals of its type in the industry and immediately established SASI as FEA Q.A. leaders. The current manual contains more than 150 basic engineering problems, each with problem description, model sketch, ANSYS input, and comparison of ANSYS output with a theoretical solution.

THE APPLICATIONS ARE LIMITED ONLY BY YOUR IMAGINATION.

But the "quality" of ANSYS extends beyond just our Q.A. verification procedures—it also involves the very nature of the program. ANSYS is intentionally written as a "general purpose" FEA program with capabilities so flexible and all-encompassing that it can be tailored to almost any application involving finite elements. In fact, engineers, scientists, and researchers are coming up with new uses for ANSYS all the time—even uses we never thought of. Because with ANSYS, you're not constrained by the limited capabilities of a "special application" program. The only limits are those of your own imagination.



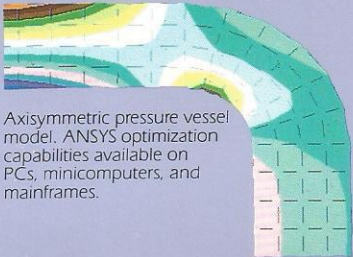
SASI's modern headquarters

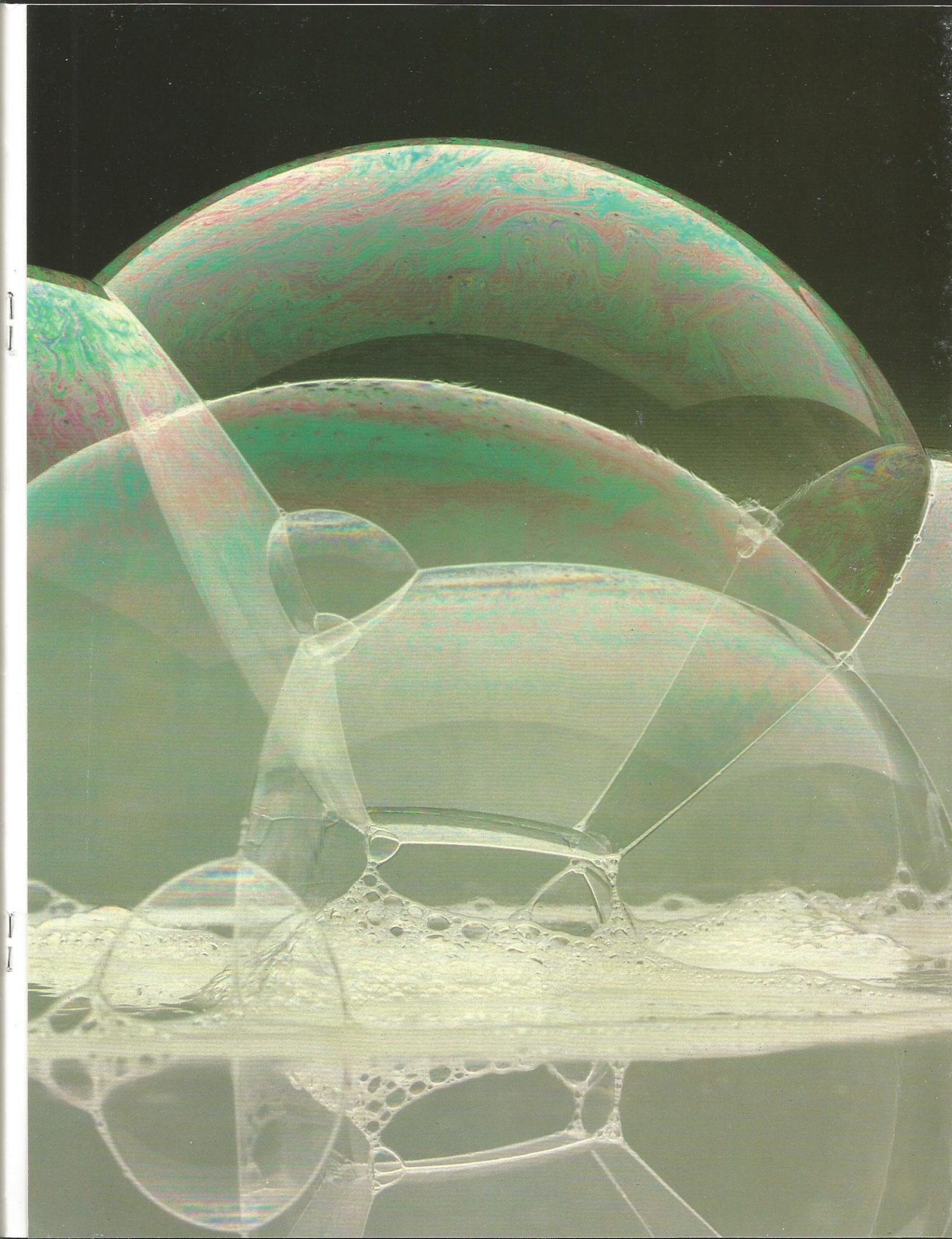
FULL-CAPABILITY FEA FOR OPTIMIZED DESIGN AND EVALUATION.

You probably blew bubbles when you were a child and watched happily as they drifted away lazily on the air. You may have marveled at the swirling crazy-quilt patterns they formed as they decorated the surface of a still pond or rushing stream. But many people still think of bubbles as nothing more than beautiful and delicate objects of whimsy.

The truth is, though, bubbles are something much more complex. They are natural pressure vessels. And they are an optimum design for the job they perform.

Optimizing man-made designs (such as pressure vessels) is a problem that's plagued engineers for years. But using ANSYS makes it much easier. Among its myriad capabilities, ANSYS features new design optimization techniques. Based on specifications established by the engineer, ANSYS can automatically determine the values of the design variables which produce the best possible design. Since ANSYS is a general purpose FEA program, the design optimization capabilities can be used for any element type or analysis procedure available. What's more, ANSYS offers easy problem definition and manipulation with features that include parametric input, solid modeling and automatic mesh generation, and pre and postprocessing databases. In short, ANSYS' design optimization gives you the tool to help produce the best possible design in the shortest time.





ANSYS CAPABILITIES.

Perhaps the greatest advantage ANSYS offers its users is its incredibly wide range of capabilities. ANSYS not only provides all the functions that engineers and scientists expect in an analysis program—such as a variety of analysis types, material representations, and a comprehensive library of elements—it offers much more. Some of the extras you might not expect include pre and postprocessing, on-line documentation, design optimization, solid modeling, and 3-dimensional graphics.

The following analysis capabilities are available in ANSYS:

Statics—Solves for the displacements and stresses in a structure subject to applied loads.

- Linear Statics assumes that the structural equilibrium equations and material properties are linear.
- Nonlinear Statics allows for nonlinear behavior in both geometry and/or material. Large deflections, stress stiffening, plasticity, creep, and interface (gap) conditions can be included in an analysis.

Dynamics—Determines the time history response of a structure to transient loading conditions.

- Linear Transient Dynamics efficiently solves for the response of a linear structure.
- Nonlinear Transient Dynamics incorporates material, geometric, and/or interface nonlinear effects.
- Spectrum Analysis envelops the dynamic response to random vibration or seismic loading.
- Harmonic Response determines the steady-state response of a linear structure subjected to harmonically time-varying loads.

Mode Frequency—computes the natural frequencies and associated mode shape of a linear, undamped structure.

Stability Analysis—Determines the load capacity of a structure.

- Linear Eigenvalue Buckling determines the critical loads (bifurcation points) and the associated buckled shapes for a linear structure.
- Large Deflection analysis determines the limit load, whether failure occurs by bifurcation or snap-through buckling.

Heat Transfer—Solves for the temperature distribution and heat flow within a body. The temperature results can be used as boundary conditions to a structural analysis.

- Linear Heat Transfer assumes non-temperature-dependent material properties.
- Nonlinear Heat Transfer can have temperature-dependent material properties (including phase change effects) and can include radiation and temperature-dependent convection boundary conditions.
- Transient Heat Transfer solves for the time-dependent temperature distribution within a body. Nonlinear effects may be included.

Magnetostatics—Solves for the intensity and flux density of a magnetic field due to current sources and/or permanent magnetic materials.

Coupled Field Analysis—Simultaneously solves for interacting multiple field effects including two or more of the following: structural displacements and forces, temperature and heat flows, electrical voltage and current, magnetic intensity and flux, and confined fluid flow pressure and velocity.

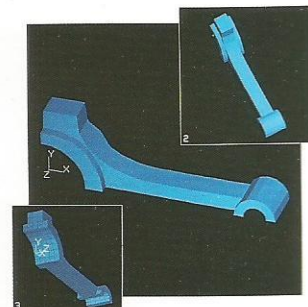
Global/Local Modeling—Used to improve the running logistics or modeling efficiency of the previously-mentioned analysis procedures.

- Substructuring physically portions out a piece or multiple pieces of a model and generates corresponding substructures for use in the full analysis—at substantial cost savings.
- Submodeling uses the results of a coarsely-modeled structure to obtain an accurate solution of a locally-refined submodel.

MATERIAL REPRESENTATIONS.

A wide variety of material representations are possible with ANSYS. Material properties may be temperature-dependent, isotropic, orthotropic, or anisotropic. Nonlinear material behavior such as plasticity, creep, swelling, and nonlinear elasticity are available in both static and dynamic analyses. Plasticity material options include:

- The von Mises yield criterion coupled with kinematic hardening which represents most metal behavior very well in the plastic range.
- Anisotropic Plasticity which allows for different stress-strain behavior in different directions as well as different behavior in tension and compression. This option can be used for composites or metals in which the yield strength is affected by processing (i.e. rolling).
- Drucker-Prager in which the material strength is dependent on the confinement pressure, such as for granular materials (i.e. dry soils, rock).



Half-symmetry model of a connecting rod. The loading was a bearing pressure which caused the rod to bend.

THE ANSYS ELEMENT LIBRARY.

The ANSYS program contains a library of more than 70 different elements. In addition, many of the elements have options which allow numerous other possibilities to be used in an analysis. The element library includes:

- Structural Elements such as spars (trusses), beams, piping, membranes, plates, thin and thick shells, layered shells, conical axisymmetric shells, axisymmetric solids, 2-D and 3-D solids, and 3-D solids with reinforcing materials. Inertial, thermal, stress stiffening, and large rotation effects are included with these elements.
- Heat Transfer Elements including conduction, convection, and radiation links; thick and thin shells; 2-D, axisymmetric, and 3-D solids. Thermal mass and heat generation effects are included in these elements.
- Thermal Electric Line Elements including 2-D, axisymmetric, and 3-D solids as well as magnetic-thermal-electric solid elements.
- Other elements include matrix elements, substructures, control elements, and immersed pipe or cable elements.
- Other effects which can be modeled include gaps, friction, viscous dampers, cables, plastic hinges, crack tips, fluid and thermal flow in pipes, and nonlinear force-deflection curves.

PRE AND POSTPROCESSING.

One of the main advantages of the ANSYS program is the integration of the three phases of finite element analysis—preprocessing, solution, and postprocessing. No additional software package is needed for generating a model or examining the analysis results. This synthesized approach means that ANSYS is fast and easy to use.

ANSYS' preprocessing routines define the model, boundary conditions, and loadings from a minimum amount of input. Displays may be created interactively on a graphics terminal as the data is developed to assist with model verification.

Postprocessing routines are used to evaluate results of the analysis in a variety of ways. They can make contour maps, display deformed shapes, perform additional calculations, and sort and select results for reports.

SOLID MODELING AND MESH GENERATION.

Until recently, the geometry and finite element mesh of an FEA model have been defined simultaneously. Now, however, ANSYS contains a versatile and sophisticated solid modeler which creates the geometry of a part or structure independently of the finite element mesh. With ANSYS, this portion of an FEA analysis requires less of the analyst's time.

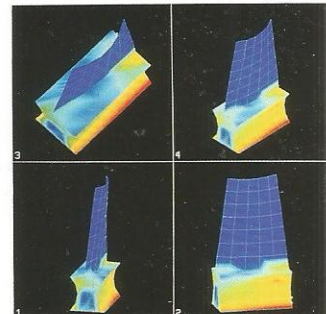
The analyst describes the model region as a volume which is bounded by areas, line segments, and keypoints. Line and area fillets, splines, solids of revolution, sweeps, and intersecting surfaces are just a few of the options available. Once this geometry is defined, a single command automatically generates nodes and elements. If necessary, the model can be changed easily and the mesh regenerated.

ANSYS INTERFACES WITH CAD SYSTEMS.

Although ANSYS contains a powerful solid modeler, the program also interfaces with a number of CAD programs. This ability makes ANSYS an even more valuable tool in the computer-aided engineering field. ANSYS features an auxiliary routine that contains a number of data translators. These translators convert a coded output file from various CAD programs to ANSYS preprocessing input commands.

PROGRAM	COMPANY
ANVIL™	Manufacturing & Consulting Services, Inc.
AUTO-TROL®	Auto-Trol Technology
FEMGEN	FEGS International, Ltd.
GRAFTEK	Graphics Technology Corp.
IGES	U.S. Dept. of Commerce, Nat. Bureau of Stds.
MEDUSA™	Prime Computer, Inc.
PATRAN™	PDA Engineering
SUPERTAB™ (Rev. 7)	General Electric, CAE International

Additionally, some CAD programs are designed to write ANSYS preprocessor commands automatically. These include Computer-union's CADDs FEM and McAuto's UNIGRAPHICS program, among others.



A turbine blade was modeled using 3-D solids for the hub and 3-D shells for the blade. Pressure loading on the blade caused bending.

PHOTO CREDITS:

1. James B. Koeneman, Harrington Arthritis Research Center
2. James B. Koeneman, Harrington Arthritis Research Center
3. James B. Koeneman, Harrington Arthritis Research Center
4. Steiger Tractor, Inc.
5. Eric Donaldson, Kensinger Associates, Inc.
6. Albert Ting, American Medical Optics
7. Albert Ting, American Medical Optics and Vigen Ghazarian
8. Joseph Bentz, Cummins Engine Company, Inc.
9. Marc Whittlesey, Analysis & Design Application Co., Ltd.
10. Marvin Johnson, Fermi National Accelerator Laboratory
11. Marvin Johnson, Fermi National Accelerator Laboratory and Dale Ostergaard, Swanson Analysis Systems, Inc.

DESIGN OPTIMIZATION.

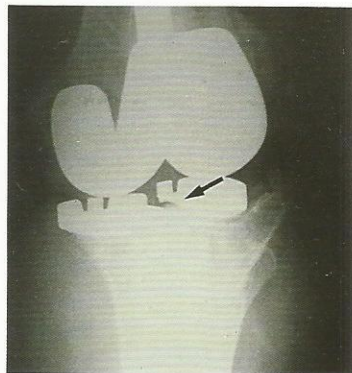
Until recently, finite element analysis has been used almost exclusively for the analysis of a user-designed model. With the new design optimization capabilities of ANSYS, however, the computer itself can now be used as an active participant in the design of a model. This technique enables the computer to generate a series of designs which are checked for feasibility, and which improve as the series progresses until a "best" design is obtained. Shape optimization, as well as optimization of parameters such as area, moment of inertia, and thickness, is possible.

PARAMETRIC ANALYSIS DEFINITION.

ANSYS fully supports parametric input, enabling the user to define a problem in terms of named variables (parameters) rather than fixed numeric quantities. This allows efficient re-modeling if the design is modified, and provides a problem definition that is suitable for ANSYS design optimization or "what if" design changes. ANSYS has other unique input capabilities such as arithmetic parameter manipulation, logical "if" branching, "go to" commands, and input subroutine calls (macros) with parameters passed. That means you can write a file of ANSYS commands and then use those commands in other analyses. In fact, the possibilities available using these capabilities are bounded only by the needs and ingenuity of the user.

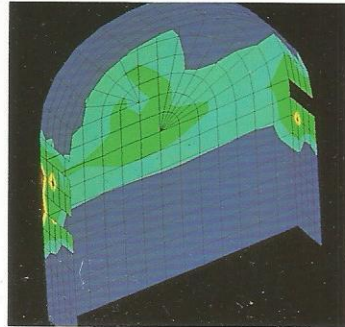
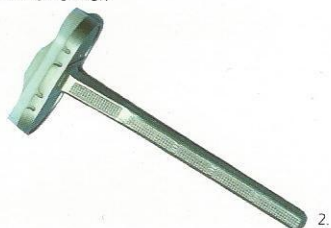
ANSYS APPLICATIONS.

Possible applications for ANSYS are as widely varied as the capabilities of the program itself. To date, ANSYS has been used to help design and evaluate everything from aircraft components, automobile engines, and suspension bridges to medical prostheses, electronic equipment, and running shoes. In other words, the applications are almost limitless. Here are a few particularly interesting examples.



1. TESTING AN IMPOSSIBLE SITUATION.

When Harrington Arthritis Research Center, a large biomedical engineering research lab, wanted to understand the causes of failure of a knee prosthesis used in treating especially severe cases of arthritis, they faced a real problem: how to evaluate a component implanted in a human body? Multiple prototype testing was impossible, yet an accurate analysis was imperative. In this difficult environment, the lab found ANSYS was their answer.



3.

The researchers set up and performed a detailed stress analysis of the metal and plastic prosthesis to determine where its weak points might lie. ANSYS enabled them to do this in a fraction of the time it would have taken to build and test prototypes. The detailed stress analysis display ANSYS produced showed them exactly where redesign of the prosthesis was necessary to reduce stress concentrations and average stress values.

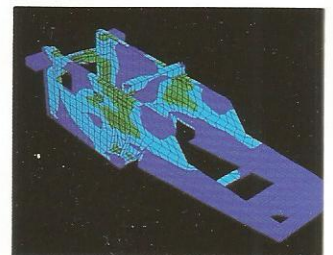


4.

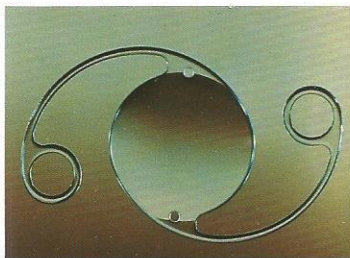
CARVING THE FAT OUT OF FARM MACHINERY.

Today's farm machinery must not only do the job for which it's designed, it must be rugged, affordable, and economical to run, too. So when Steiger Tractor, Inc. needed to design chassis frames for a new, cost-effective tractor model, they turned to ANSYS.

Using linear static analysis with substructuring, they were able to design the frames using less and lighter-weight material—without giving up essential strength or rigidity. That cut their production costs and helped them keep prices in line. And it gave the farmers a lighter tractor with greater fuel economy and reduced soil compaction.



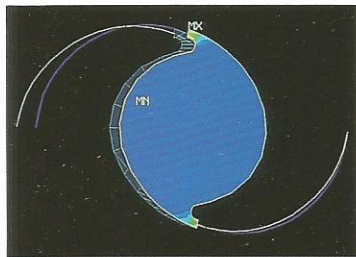
5.



6.

BRINGING NEW SIGHT TO CATARACT PATIENTS.

American Medical Optics used ANSYS in the design of a revolutionary product called the intraocular lens (IOL) which replaces a human eye lens removed during cataract surgery. The IOL is made of acrylic plastic and consists of a circular, disc-shaped "optic" portion and cantilevered, arch-shaped supporting "haptics." After implantation, the IOL is held in place by the resilience of the partially-deflected haptics.

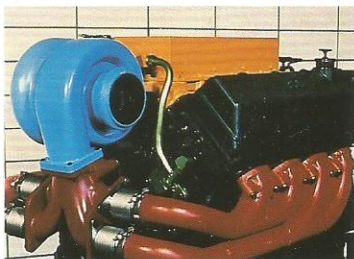


7.

ANSYS analysis was used to find the balance between maximum stress in the IOL material and the minimum pressure exerted on the ocular tissue by the haptic. A preliminary IOL was modeled and tested for flexibility using ANSYS' large deflection analysis. Stress and deflected geometry information was used to assist in the IOL's final design.

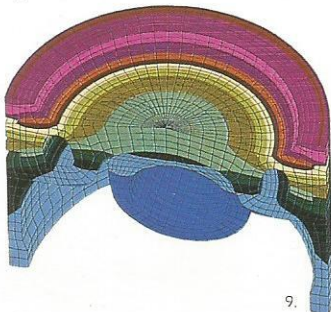
TAKING THE HEAT OFF DIESEL ENGINES.

A major manufacturer of diesel engines, Cummins Engine Company, Inc., had to find a way to cool only an engine's head and pistons instead of the entire engine block. With ANSYS, they were able to achieve a satisfactory prototype and subsequent product without the expensive and time-consuming procedure of actually building and rebuilding various test models.



8.

This company used ANSYS to model and test the experimental engine on their computer. In order to determine where significant thermal stresses occurred and where maximum cooling was necessary, heat transfer analyses were performed. The analysts also evaluated how thermal stresses affected the engine's performance and longevity. Various materials were compared using ANSYS to evaluate how they would stand up to the severe duty proposed. The data ANSYS generated is now being incorporated into the design of what may someday become a new generation of lighter-weight, higher-performance diesel engines.



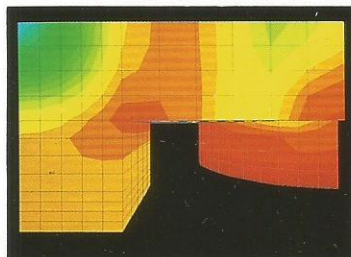
9.



10.

EFFECTIVE PARTICLE ACCELERATOR DESIGN.

Few FEA programs are capable of performing large-scale electro-magnetic analysis—but ANSYS is. Recently, researchers at the Fermi National Accelerator Laboratory built a superconducting magnet for use in a sub-atomic particle accelerator. They used ANSYS to help them find how effectively their accelerator was able to obtain the desired magnetic field under operational conditions.



11.

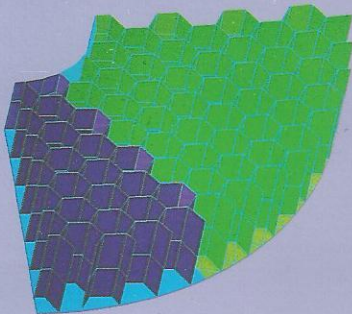
After first supplying relevant operational data to ANSYS, plots generated were used to visualize the flux density distribution in the air gap between the poles of the magnet and in the space surrounding the magnet structure. These analyses enabled the researchers to predict the desired magnitude and uniformity in flux density between the magnetic poles and determine the degree of saturation in each of the magnet's "legs."

ANSYS LOOKS INTO NEW FIELDS.

The eye of an insect represents a totally different way of seeing. Rather than a single lens which creates a single image on a focal plane, the insect's eye incorporates thousands of individual hexagonal facets to create a mosaic-like image which is interpreted by the insect's brain. The more facets the insect's eye has, the finer the mosaic. The combined focusing and light-gathering power of these multiple facets lets the insect use its eyes much like a powerful telescope. It can detect the slightest movement of an enemy at long range—or zoom in for a close-up on a potential meal.

Today, a new type of telescope which uses a similar array of hexagonally-shaped cells to support a 2-ton, 140-inch diameter mirror is under development. Lighter in weight and less expensive than traditional telescope mirrors, this mirror assembly rests atop a set of adjustable air pistons used for support and collimation. Scheduled to go into operation in 1988, the telescope will be located at Apache Point, New Mexico, but will be remotely operated by astronomers at five collaborating universities. It will be able to probe the structure of the universe 15 billion light-years into the night sky.

And once again, ANSYS helped make a technological breakthrough possible. ANSYS was used to investigate the critical deflections of the reflective surface by modeling the mirror assembly and subjecting it to its own weight and the air piston forces. Several deployment patterns for the air pistons were analyzed to determine which minimized surface deflection. Focal surface degradation caused by unequal thermal operating loads was also investigated. Just one more demonstration of the wide range of ANSYS capabilities.



Telescope mirror support structure.



WHY CHOOSE ANSYS?

Fifteen years ago if you wanted to use FEA, there were only a few programs available. Today there are dozens. So why is ANSYS still your best choice? Of course, there are the obvious advantages of the ANSYS program's general purpose format and full-capability performance that make it useful in almost any finite element application. But there are also a number of other excellent reasons that make ANSYS stand out from the crowd.

ANSYS RUNS ON MOST MAJOR BRANDS OF HARDWARE.

To offer the advantages of ANSYS to the widest possible range of users, the program can be leased on the following computers:

Alliant	Floating Point
Amdahl	Systems
Apollo	Fujitsu
Cyber	Harris
Celerity	Hewlett-Packard
Computervision	IBM
Convex	Prime
Cray	Ridge
Data General	Sun
DEC	Univac
ELXSI	

Of course, ANSYS doesn't run on every type of computer and this list will change from time to time. But as new and more advanced computer hardware is introduced, SASI will make sure that ANSYS keeps pace by continually developing the program to take advantage of the latest technological innovations.

ANSYS IS AVAILABLE ON PC'S, TOO.

Even if you don't have access to a mainframe, you can still have the advantages of using ANSYS. A family of PC products, all rigidly-defined subsets of the full ANSYS program, are available to you as well. These programs are designed to operate on personal computers such as the IBM®PC/XT, PC/AT, or similar compatibles.

ANSYS®-PC/LINEAR, the most popular of these PC programs, is a complete program including pre-processing, solution, and post-processing. It can prepare, solve, and evaluate linear static and modal analyses. The program is capable of comprehensive analyses using its extensive element library, and its model input data can be passed on to ANSYS for analyses on larger computers. ANSYS-PC/LINEAR fills the need for accessible, affordable linear analysis.

EDUCATIONAL VERSIONS OF ANSYS.

SASI distributes two inexpensive versions of ANSYS. They offer an easy method of providing extensive finite element analysis experience to industries and educational institutions.

The ANSYS University Version has the same capabilities as the full program except that the allowable problem size is limited. Available to colleges and universities, this educational version of ANSYS is to be used for student instruction, projects, and demonstrations. It is currently in use in more than 200 educational institutions.

ANSYS-PC/ED is a two-dimensional version of ANSYS-PC/LINEAR which can prepare, solve, and evaluate linear static and modal analyses on the IBM PC/XT, PC/AT, and compatibles. It is available for educational use only to industries, colleges, and universities, and is ideal for training or demonstration.

WE SUPPORT ANSYS LIKE NO OTHER FEA PROGRAM.

Because support services can be absolutely crucial to a user's satisfaction with any engineering computer program, Swanson Analysis Systems, Inc. provides support for ANSYS unsurpassed by any competitor.

To start with, we're not a diversified company. ANSYS is our only product and our only business purpose is to distribute, maintain, and support the ANSYS program.

SASI provides complete training in the use of ANSYS. A three-day introductory seminar provides users with a comprehensive overview of ANSYS capabilities and familiarizes them with operational techniques. We also offer a continuing program of advanced training seminars based on user requests. And we're adding new seminars on new subjects all the time to keep pace with the ever-changing face of FEA.

We provide extensive, comprehensive documentation for ANSYS, too. Manuals available include our User's Manual, Examples Manual, Verification Manual, Theoretical Manual, Introductory Manual, Seminar Notes, Command Guide, and a Techniques Manual. No other software supplier provides such complete documentation for FEA use.

We operate a network of ANSYS Support Distributors throughout the world to provide service and help users solve their problems. We can even provide special consultants, on a fee basis, to assist with especially difficult problems.

Last, but certainly not least, SASI has a full-time Customer Support staff at our headquarters and our distributor locations. At these locations we maintain an ANSYS hotline so you can contact us directly when you need help in a hurry.

HOW CAN ANSYS HELP YOU?

Since 1970, Swanson Analysis Systems, Inc. has been committed to just one goal—providing ANSYS users with the most reliable and innovative FEA program backed by the highest quality level of support. Coupled with the latest hardware, ANSYS gives you the most powerful and sophisticated FEA technology available. Plans are already on the drawing board for revisions which will be released years from now. Yet while we look to the future, we never lose sight of the ANSYS users of today. That's why ANSYS is the answer to your design and analysis problems.

IBM®PC/XT and PC/AT are registered trademarks of International Business Machines Corporation.



SWANSON ANALYSIS SYSTEMS, INC.
Johnson Rd., P.O. Box 65, Houston, PA 15342-0065
TWX: 510-690-8655 Phone: (412) 746-3304